LOCKHEED MARTIN GPS SATELLITE BRACKET – SOFTWARE TEAM

Paul Chancellor, Drew Leadford, Sam Piltz, Drew Stafford

Abstract

Design software packages have become more advanced and dynamic, while 3D printing has become more accessible and commonplace; however, the best practices to marry the two are not completely understood. Our guiding ethos was to use and evaluate two top performing topology optimization software packages, Ansys and nTopology Platform, for use in computer-driven design and additive manufacturing. We had initially planned to examine a third package, Simufact, but upon further evaluation determined that it is geared primarily toward printer optimization rather than design, so it was excluded in favor of more extensively studying Ansys and nTopology. Our goal was to create two types of bracket designs testing the limits of these software packages and to make a recommendation to Lockheed Martin regarding which software was better tasked for each type of design.

The two software packages are both capable of creating computer-driven design. The nTopology's Platform software utilizes topology optimization, which is the process by which an existing or traditional part design is optimized – typically by removing excess material to optimize performance for given use conditions. Ansys, by contrast, utilizes generative design, by which a volume and set of boundary conditions are defined, and the part design is completely generated by the computer. Philosophically, topology optimization improves on an existing design, while generative design can create entirely new and organic structures. Both approaches are able to take advantage of complex geometries that are only made possible through additive manufacturing.

Using the topology optimization approach, first a series of rectangular pieces were designed to construct a box-like bracket. Stresses were applied to the structure to reveal on optimized truss structure for the side pieces of the bracket. With the current design meeting all design requirements, we were then able to introduce lattice structures, which gave us a marked increase in stiffness, while adding relatively little weight, all the while eliminating the need for support material.

Using the generative design approach, we created a large volume of space surrounding the mounting points of the rocket motors. After applying the boundary and loading conditions to the volume, we set a mass requirement of seven pounds. The end result was a single piece design with organic structures. The design was augmented with a diamond lattice along the base of the rocket housings to eliminate the need for support material.

Both designs have benefits and found both software packages to be powerful tools for engineers. During the process, we creating a ranking table measuring attributes that include ease of learning, software input time, repeatability of workflow, and more. In the end, based on our ranking criteria, we concluded that Ansys should likely be the software for Additive Manufacturing parts for Lockheed Martin.

Overview

Our team was tasked by Lockheed Martin to redesign a bracket for the GPS satellite. We were given design specifications for the current bracket and told to design something new in the space. The goal was to create design that could take advantage of the new ability to 3D print aircraft grade aluminum parts.

This led us to looking at new methods of design. While traditional methods are certainly valuable, AM creates opportunities for incredible innovation in design. To create these new designs, the simple design motifs and guidelines that we often follow would not be sufficient. Instead, we wanted to utilize new software packages, capable of creating optimized parts that are not restrained by traditional manufacturing methods.

The two methods we were most interested in were generative design and topology optimization. In topology optimization, a part is first constructed in the CAD program. The software then uses loading conditions, boundary conditions and material properties to iteratively remove material until only the crucial elements are left. This model can then be smoothed and finished by the engineer, with new features added whenever the user desires, including options for removing specific material for particular purposes (such as the formation of lattices).

In generative design, these same inputs are placed into a design space, and the software organically creates a form and shape that can meet those design requirements. This often leads to more ‘radical’ designs, as the computer is not restrained by any kind of traditional model to begin with. Many consider this to be a large part of future design, whereas topology optimization is seen by many to be the bridge between current design and this radical, computer driven design of the future.

To attempt both of these new design techniques, we chose to try both Ansys (a powerful design software with a lot of simulation built in, that specializes in generative design) and nTopology’s Platform (a current top tier topology software). We decided to pursue two simultaneous designs - one more conservative design with the more conservative nTopology, and a more aggressive, new generative design with Ansys. By going through this process, we could learn not just about which individual software is easier to use and more polished, but the process would also allow us to gauge which approach Lockheed Martin finds more use in, as the preferred design would likely pair with the preferred package.

Both of the designs will now be explained by the primary creators: Paul Chancellor, who designed the conservative bracket with nTopology, and Drew Stafford, who designed the generative form with Ansys.

Conservative Bracket Design using SolidWorks and nTopology Software for Topology Optimization

The conservative bracket design made use of two different software platforms, Dassault Systems SolidWorks and a newcomer in the industry, nTopology. SolidWorks was utilized for CAD modeling and baseline Topology Optimization. Each of these platforms offers the ability to perform Topology Optimization, but nTopology does not have a native CAD capability. This is known and offered unapologetically by nTopology. They intend their software to be used in the additive manufacturing (AM) space primarily. In this area of technology, it shines a very bright light. The intent was to highlight the benefits of nTopology because the sponsors of this project were interested in this.

The key project knowns as listed by LM were as follows:

1. The assembly had 2 rocket engines that are 1.7 [lbm] and produced 5 [lbf] each.
2. Seven 6061-T6 aluminum traditionally manufactured parts.
3. Twenty-six #8 fasteners joining components.
4. Two to four service valves that could exhibit up to 250 [in-lb] torque.
5. A total component mass of 6.1 [lbs].

The design constraints were given as follows:

1. Assembly with masses must withstand a 25-g mass acceleration curve in all three individual axes.
2. The natural mode of vibration must be greater than 100 [Hz].
3. Hardpoint mounts to the satellite bus must be kept from current design.
4. Hardpoint mounts for components must be kept from current design.
5. Fastener inserts loads must be kept to 442 [lbf] for pullout and 348 [lbf] for shear.
6. Must have a safety factor (SF) of greater than 1.25 for yield with an ultimate SF greater than 1.4.
7. Clearance for all wiring and tubing bundles must be maintained and exit the area opposite the valve mount face.
8. Thermal isolation spacers for rocket engines must be incorporated into design.
9. Heater footprint behind service valves must be maintained.
10. Environmental temperature range of -60C to +60C must be considered.

To begin the process of optimization, a model was needed to optimize. The design space dimensions were provided by drawings from LM and used to create a simple geometric bracket assembly. The model has three regions with the following naming convention: Rocket Engine Bracket - which are the brackets that provide the rocket engine interface; Side Frame Support Bracket - A middle region where bracket supports are to give a mounting surface for the rocket brackets; Front Valve Plate - the plate where the service valves mount. The model is seen here in Fig. 1.

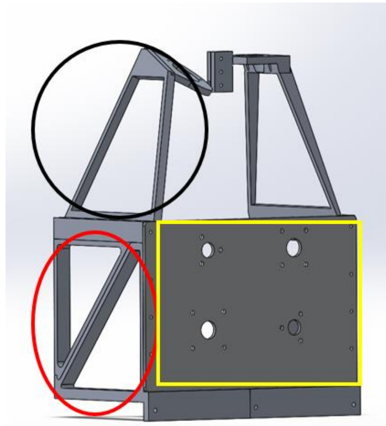


Figure 1*: *Model 1.

Model 1 has no Finite Element Analysis (FEA) applied to the design and has simple bonded joints to make up the assembly. The dimensions of all parts, except for the Front Valve Plate, are sized to fit in the build space of the EOS M290 metal additive manufacturing machine. By setting the dimensions up this way all future modifications can now be produced using traditional subtractive manufacturing methods or AM methods. This process ensures flexibility for LM.

The first update of Model 1 sets up the conditions necessary to apply FEA. This is important because Topology Optimization (TO) is built on FEA. Specifically, the computer software uses the criteria for allowable stresses or displacements to compute where it is acceptable to remove material or mass. To set these conditions up, components with mass were added to the assembly and fasteners were placed in the joints. This is illustrated in Fig. 2.

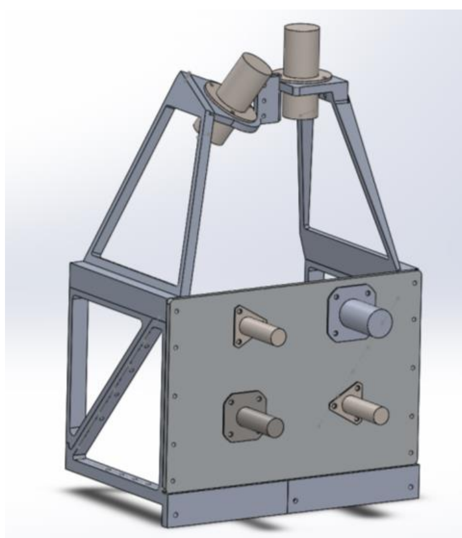


Figure 2: *Model Rev 1 for FEA.*

One of the first applications of FEA for this updated design was a static analysis where a 25 g load was applied to all three axes of the model. The 25 g load was provided by LM and is based on launch loads experienced by the spacecraft. This was to ensure that the design was sound for future modifications. The results of the FEA static studies are shown in Table 1, as well as in Fig. 3, 4, and 5. The material used for analysis is Al 6061-T6 and is part of the required specifications for the project. Note that the Factor of Safety (FOS) is greater than 1 for all three loading directions, based on allowable stresses and the material properties of the aluminum. The stress results in Fig. 3-5 verify these FOS results.

Table 1: *Results of Static Analysis First Assembly*

Axis	**X**	**Y**	**Z**
FOS	>1.75	>1.75	>1.4

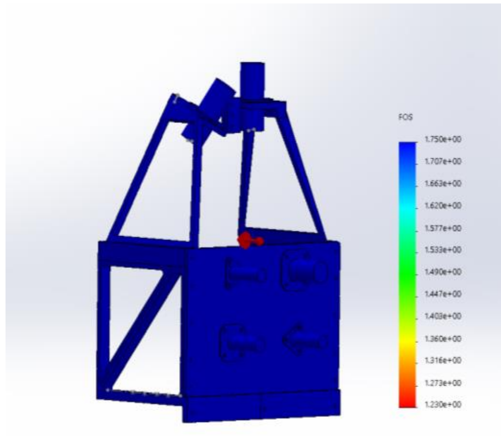


Figure 3: *X Axis Static Factor of Safety Analysis.*

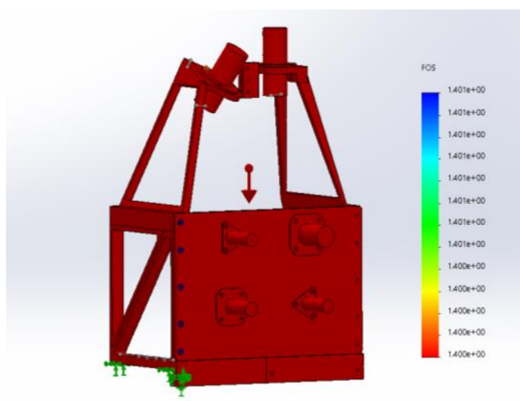


Figure 4: *Y Axis Static Factor of Safety Analysis.*

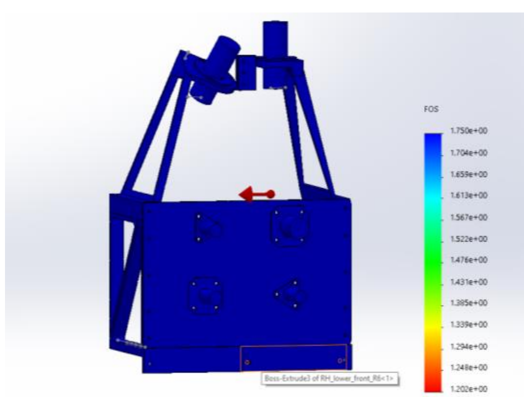
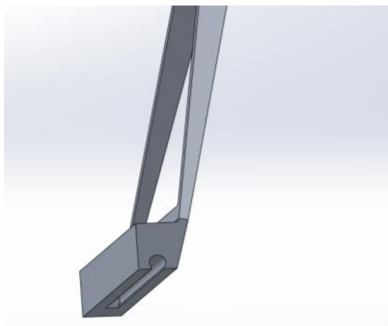


Figure 5: *Z Axis Static Factor of Safety Analysis.*

Now that Model Rev 1 had been tested for validity, the plan was to start exploring different potential design changes for enhancement. Bob Rauscher, product design manager for LM, expressed a desire to focus on reducing fastener count in the assembly. This was considered of importance due to significant man-hour requirements in assembly with fasteners. As an approach to minimize the fastener count, the idea of precision loaded joints was explored. This concept is prevalent in Japanese carpentry. The loaded joint theory minimizes fasteners as the parts become stabilized by the stress-induced bond at the joint interface. Figures 6 and 7 show an example joint that was considered.



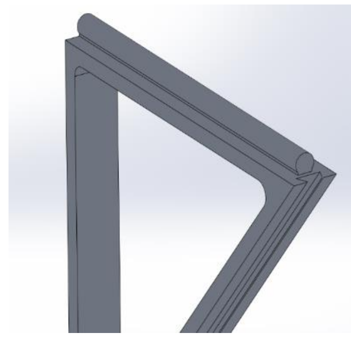
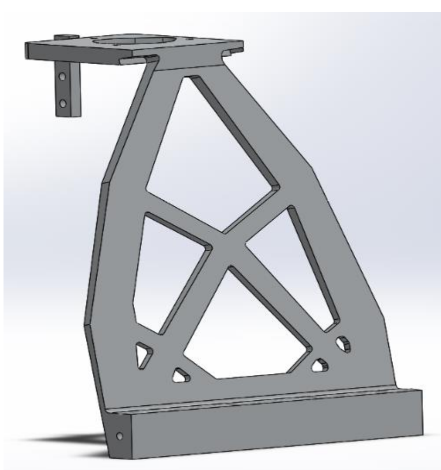


Figure 6 (left): *Curved Insert Joint Receptacle, Bottom of Rocket Engine A Bracket (Model Rev 2).*

Figure 7 (right): *Curved Insert Joint, Top of Side-Frame Support Bracket (Model Rev 2).*

While this style of connection showed promise with an acceptable safety factor after performing FEA, ultimately, it was decided to drop further study of it. One consideration was the low compressibility of metal vs wood. Another reason was the manufacturing complexity introduced by these joint concepts would possibly add further complications down the road.

The process of applying FEA and TO began with Model Rev 3. Because TO and FEA are computationally expensive, analysis studies can run several hours, the model was broken down into individual component analysis for the sake of time. TO poses an additional challenge in that it also needed to incorporate the design principles for AM. As it stands currently, no single package has these limitations for AM built in. Merging AM principles to the TO process depends on input from the operator of the software package. The procedure for TO is to run the simulation and blend the result to match AM best practices. This can give rise to parts that may not look computer-generated but meet the requirements. Figures 8 and 9 show the outcome of TO simulation for the Rocket Engine Bracket denoted as "A".



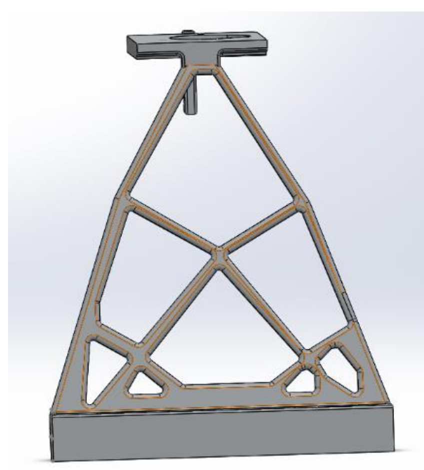


Figure 8 (left): *2nd pass of TO Rocket Engine Bracket A (Model Rev 3) guided by operator.*

Figure 9 (right): *Final pass of TO Rocket Engine Bracket A (Model Rev 6.1) guided by operator.*

Similarly, guided TO applied to the support brackets for each Side-Frame Support Bracket produced the following changes seen in Fig. 10 and 11.

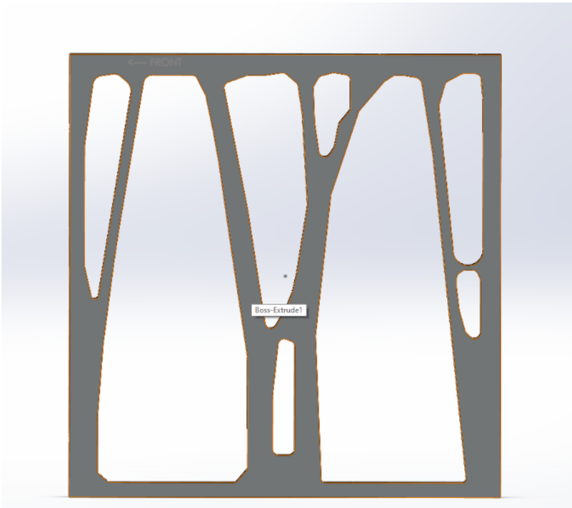


Figure 10 (left): *TO Mid-Frame Bracket (Model Rev 10).*

Figure 11 (right): *TO Mid-Frame Bracket (Model Rev 11).*

One benefit of this type of TO process is that it provides some measure of control over the results. The side support brackets in Fig. 10 and 11 were developed by the operator so that either an additive or subtractive manufacturing process could be applied. By using guided design principles, part count for the Mid-Frame Bracket region was reduced by 50%.

The final design of the Front Valve Plate after TO is seen here in Fig. 12. This result was 26% lighter and met all stress requirements.

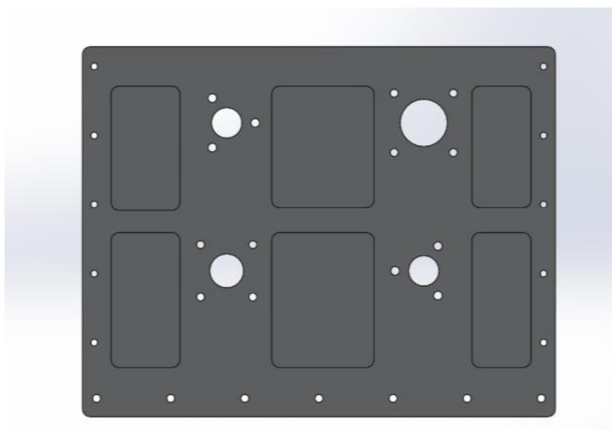


Figure 12: *Front Valve Plate (Model Rev 10.1).*

The process of TO continued until Model Rev 11. The overall mass now had been reduced by 18%. The analysis had been largely conducted in SolidWorks up to this point due to it being a more mature software package. Moving forward, nTopology would be used to produce new AM capable designs. The final model assembly is shown here in Fig. 13.

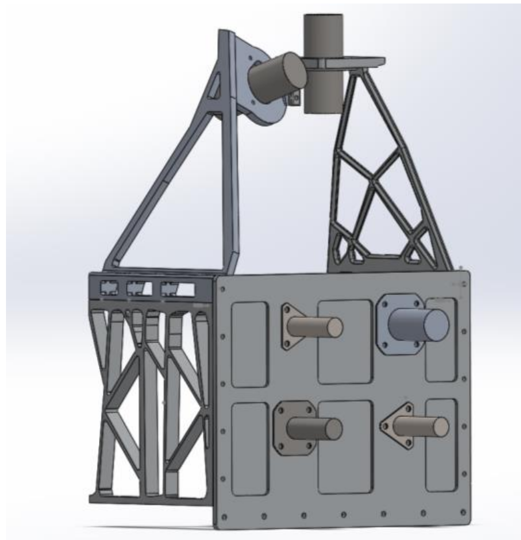


Figure 13: *Final CAD assembly (Model Rev 11) heavy TO applied.*

nTopology is a relative newcomer to the engineering software field. They are aiming to be a major player in the AM space. They have only been around since 2015 and have come a long way since the beginning of this project. They now have several ways to push learning content and have taken an active interest in user feedback. This has led to a much more user-friendly interface. nTopology brings a lot to the table because it enables designs to take advantage of AM abilities to print complex shapes. Unfortunately, the project was ramping up to start hands-on research of these complex designs when current global events pushed everyone out of the building. As such, we have a limited amount of data to provide LM in this arena.

One of the ways in which nTopology was utilized included adding light-weight lattice structures to test print capability and enhancements of parts. Figure 14 shows a cube vertex centroid lattice applied to the open space in Rocket Engine Bracket “B”.

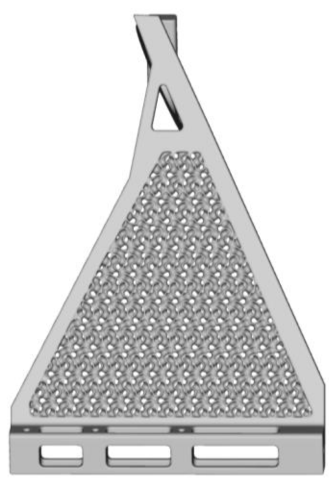


Figure 14: *Rocket Engine Bracket “B” with Gyroid lattice.*

This lattice addition added a minimal mass increase of 14% to this Rocket Engine bracket but increased the stiffness response by 34%. These results were not unusual and changing the style of lattice could be accomplished easily. The trouble could be found in the actual print of the part. The orientation of the part at times would be less than ideal for printing and result in aberrations. Some of the problems include shift lines and rough downward surface finishes. The defects would then require further post-processing if a polished product were to be expected. Striking a balance of complex lattice yielding excellent strength to weight benefits will require an experienced operator for both software and printer.

Another interesting benefit of nTopology is the ability to perform side by side differential analysis on a part. For instance, Fig. 15 shows static analysis performed on Rocket Engine Bracket B comparing the reaction of the solid body only to the reaction of the body combined with lattice addition. This capability made it easy to see gains or differences in model response.

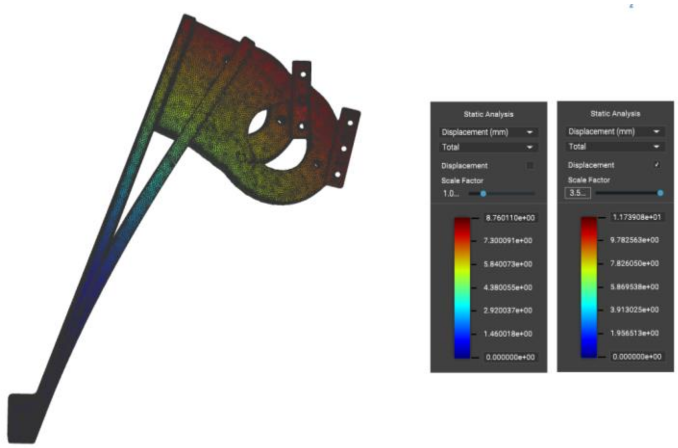


Figure 15: *Static analysis comparison on Rocket Engine Bracket B.*

These lattice additions were verified to print in polymer but must still be verified in Al 6061-T6 in the EOS M290. It is believed that they will certainly print, but possibly could require some extra fine-tuning to get the proper results.

Some other features of nTopology that were not explored is the native export and bundling of printer-specific print language. nTopology has been working with EOS and Renishaw to produce the ability to slice and export parts in the build space of multiple printers by these manufactures. It is hoped that these features would have eliminated the requirement for yet another piece of specialized software. This is another area the project was going to test but was cut short.

The overall outcome of this conservative bracket design produced an assembly that is compliant with the constraints and achieved a 14% reduction in mass, 28% reduction in part count, and a 27% reduction in fasteners required for assembly of major parts. It is believed that more research in the AM space using nTopology will give LM even greater gains in product development through hands-on printing and refinement of any designs or models.

Generative Bracket Design using Ansys Software for Generative Design

The goal of the project was to create a satellite support system bracket assembly using additive manufacturing capabilities. A generative design was chosen, allowing finite element analysis (FEA) to determine what material was necessary to support given loading conditions. A design space was created that conformed to the size allocated for the bracket assembly by Lockheed Martin (LM), and that would fit within the print volume of the M290 aluminum printer. A generative design was then produced, and features were added that would ease the challenges of printing a complex shape. This process was performed on key components that make up the entire bracket assembly. A total of 3 unique parts are used in this assembly: one rocket housing, two side pieces, and one valve plate. Each design iteration significantly enhanced the bracket system in terms of better meeting the design goals.

When building a design space there are numerous design constraints to consider. First, we want to limit the overall dimensions of the design to conform with the known constraints for size. In this case, we need to make sure the rocket housing fits the space available on the GPS III satellite. Secondly, we need to make sure this design space can fit inside the print volume of the printer that is going to be used to build the part. Once these two conditions are met, the focus can then be on additional design specifications regarding stresses, weight, part count and frequency.

When building the design space, it is important to realize from the beginning which features we need to build into the part that we will extract from the design space. Mounting locations and mounting surfaces need to be designed into the product before it is used to make a generative part. This allows these features to be selected as constraints so that they may exist within the generative part without having to later add them, which is much more difficult. In the case of the rocket housing, considerations include mounting locations of the rockets to the rocket housing, and the locations where the support structure attaches to the rocket housing. Figure 16 shows a design space used to generate the rocket housing.

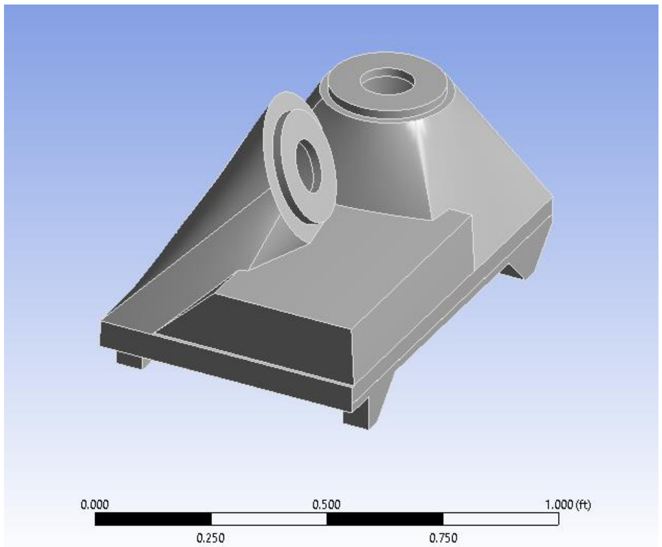


Figure 16: *Shows a design space that was created to be used to generatively design a rocket housing. The beginning weight was 15.975 pounds.*

Once a design space has been created, it will be loaded into the computer software. For this bracket assembly, Ansys software packages were used. The first step in creating a generative design is to ensure that the design space itself can withstand the loading conditions specified. This bracket utilized two 5-pound thrust rockets. A factor of safety of 5 was included on these loading conditions at the location that the rockets would be mounted. Table 2 shows the stress and strain values experienced by the design space. Figures 17 and 18 show a visual representation of the design's stress and strain under loading conditions.

Table 2: *Values for stress and strain experienced by the design space under loaded conditions*

Stress	**Minimum [psf]**	**Maximum [psf]**	**Average [psf]**	**Allowed [psf]**
	1.50E-07	1.36E+04	7.12E+03	5.76E+06
Strain	**Minimum [ft/ft]**	**Maximum [ft/ft]**	**Average [ft/ft]**	
	1.07E-16	9.21E-06	5.48E-07	

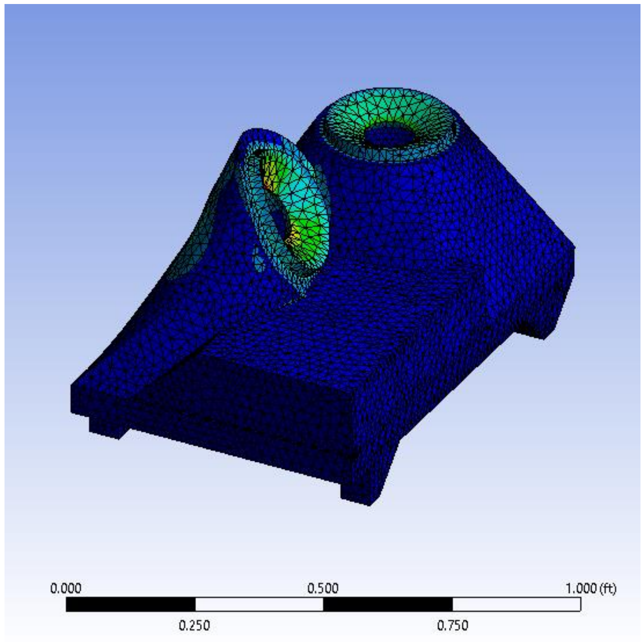


Figure 17: *Stress on the design space with 25 pounds of force applied to both rocket mount locations*.

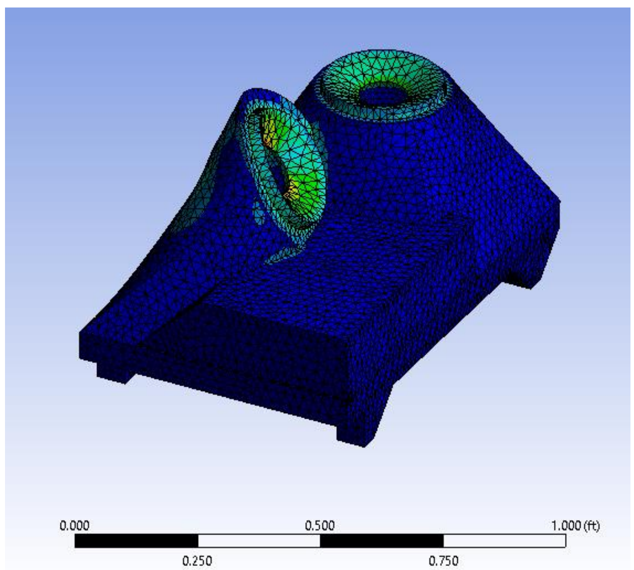


Figure 18: *Strain on the design space with 25 pounds of force applied to both rocket mount locations*.

After it is confirmed that the design can withstand the loading conditions, as validated by the finite element analysis, we perform the topology optimization that will create our generative design. We need to select between the options of mass reduction or volume reduction. In this case, mass reduction was selected, and we assigned a value of 15% to retain of the original mass. Figure 19 shows the generative design after topology optimization was performed.

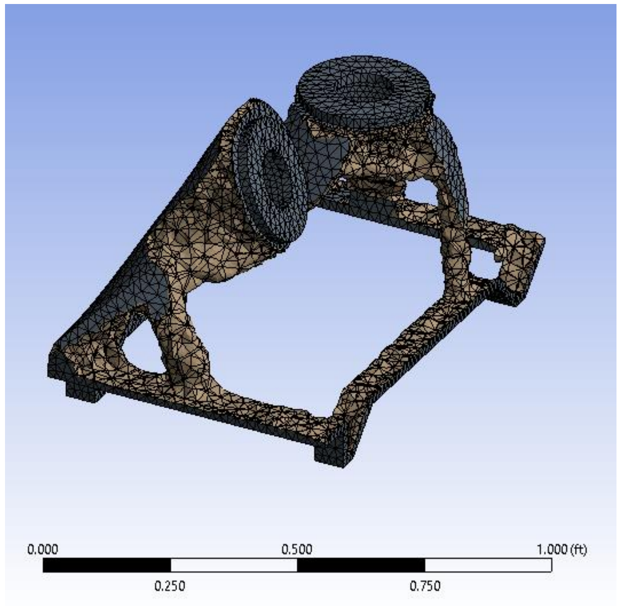


Figure 19: *Generative design after topology optimization. The design is now 15% the original mass and weighs 3.128 pounds. Note that mounting locations were preserved by choosing them as constraints. This was accomplished in the design space phase.*

At this stage, a new static structural would be performed on the part. Due to limitations existing with the academic version of the Ansys software packages a mesh could not be produced on a part this complex.

The side pieces were constructed in a very similar way to the rocket housing with a few key differences. The geometry of the side pieces was considered simple enough to eliminate all support material considerations on the EOS M290 printer. Support material is added material that the printer will add while printing the part. The support material will then support the part where angles of printing are not otherwise achievable. Therefore, these parts had a more hands on approach in their design. While the rocket housing has sweeping organic looking arms that support the load, the side pieces were made to look very traditional so that all support material could be eliminated.

The side pieces start their creation as a block of material. Figure 20 shows the design space for the side pieces.

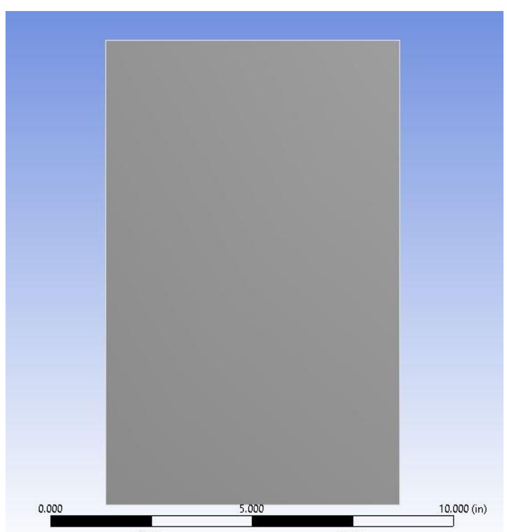


Figure 20: *A simple block is where the side pieces begin.*

After the forces of the rockets are applied to the side pieces, a finite element analysis is performed to determine the stress and strain exerted on the design. Figures 21 and 22 show the stress and strain experienced by the design space.

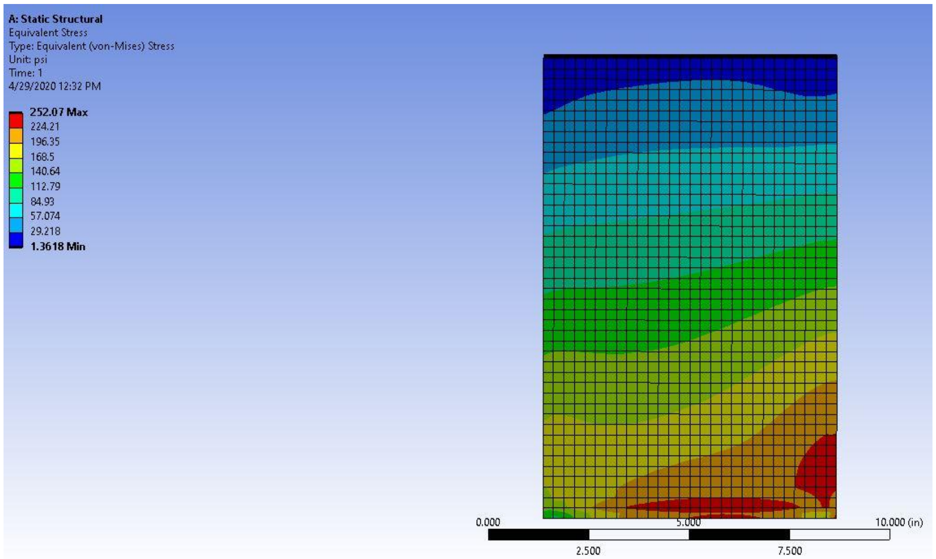


Figure 21: *Stress on the side piece design space under loaded conditions*.

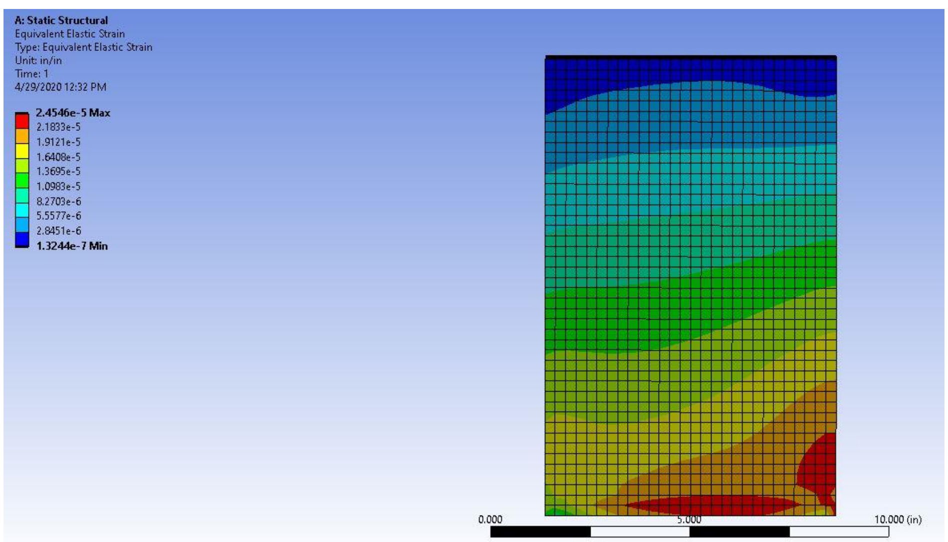


Figure 22: *Strain on the side piece design space under loaded conditions*.

After it is determined that the side piece design space can withstand the loading conditions, previously determined by the finite element analysis, the design space is optimized to reduce mass. In the case of the side pieces, a reduction of 80% of the mass was selected. To be clear, the part, after reduction, can support the loading conditions specified prior to reduction. Figure 23 shows the side piece design space after topology optimization.

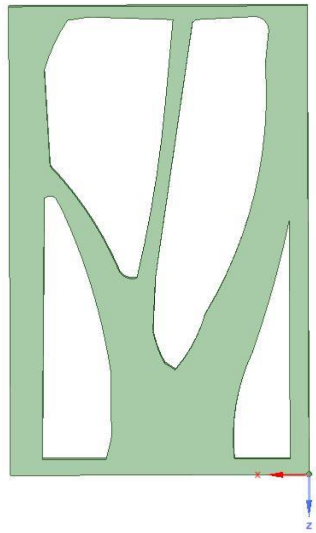
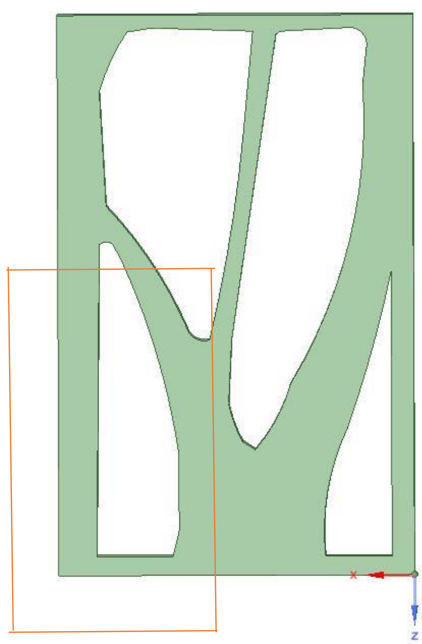


Figure 23: *Side piece after topology optimization.*

Now that we have the basic design, we want to ensure that the entire structure can be printed with no support material, while withstanding all known constraints, including a natural frequency above 100 [Hz]. A corner of the side piece was selected and mirrored about two axes. Figure 24 shows which corner was selected. Figures 24 and 25 show the mirrored side piece after it has been modified in thickness and printable angles.



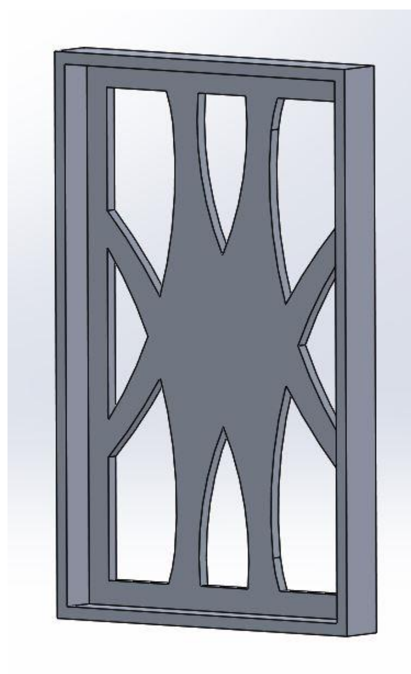


Figure 24 (left): *Shows the mirrored corner of the optimized side piece.*

Figure 25 (right): *Side piece that has been mirrored from one quadrant of the optimized side piece. This piece was then reduced in thickness in the center, and all angles were changed to make it printable with no support material.*

Now we have two parts that make up much of the assembly we can further refine them to reduce weight and make them easier to print. We begin with the rocket housing.

For the changes made to the rocket housing the information will be presented in alphabetical order that follows the revision change log (example, Reva, Revb, etc.).

The rocket housing starts as a very rough looking generatively designed shape and it is apparent it needs a tremendous amount of work to redesign it to be a functioning part. Figure 26 shows the rocket housing before any revisions.

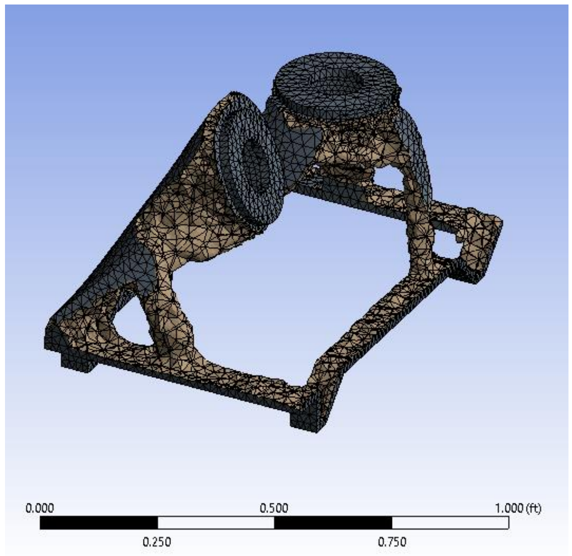


Figure 26: *Rocket housing before any revisions.*

Some initial smoothing is done in mesh mixer to see if the generatively designed rocket housing can be cleaned up a little while exposing areas that might be at risk. We can see an area that is at risk highlighted in pink below. This is where the smoothing operation eliminated too much material. This is called Rev3a. We can see in Fig. 27 that a portion on the back side of the rocket housing is missing material completely, highlighted in pink.

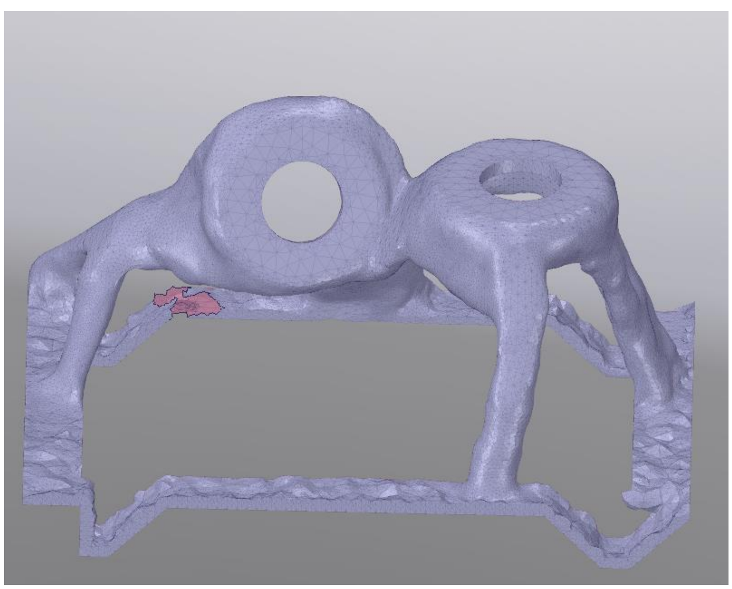


Figure 27: *Rev3a shows missing material in pink located in this picture in the middle left side*.

To correct the issue with missing material in Rev3a, material needs to be added. The mounting surfaces also need to be filled in and given a smooth geometry again. We can see this change in Fig. 28, which shows Rev3b.

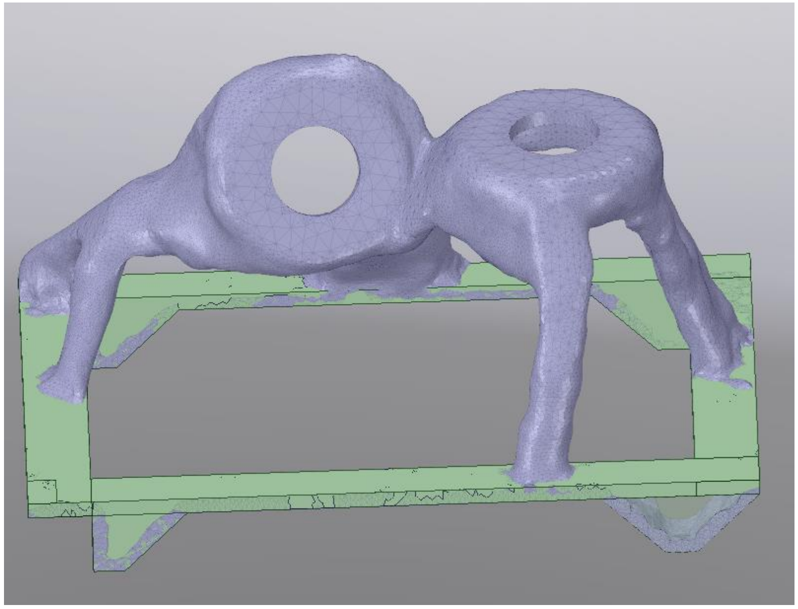


Figure 28: *Rev3b shows that the missing material has been filled in along with all mating surfaces given a smooth geometry again. All the changes can be seen in green. These changes will be combined with the model to form a single solid body.*

Rev3c adds a flat surface on the side of the rocket bracket. It is half constructed in this revision. This will make it so no support material will be required when the bracket begins printing. It will be a solid portion printed directly to the print bed. This way, the part is not built entirely on support material. A strong foundation is a good analogy for this change which can be seen in Fig. 29.

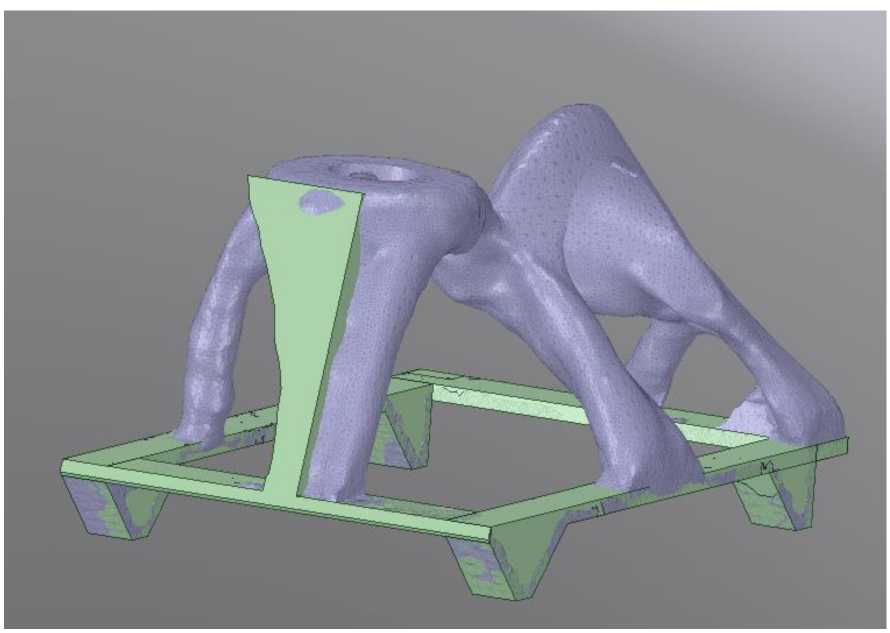


Figure 29: *Shows Rev3c with a half-built platform that will eliminate all support material at the beginning of the print.*

Revision 3d completes the platform that will allow the part to start printing with no support material, or the strong foundation. Also, these changes have been combined into one part. These changes can be seen in Fig. 30.

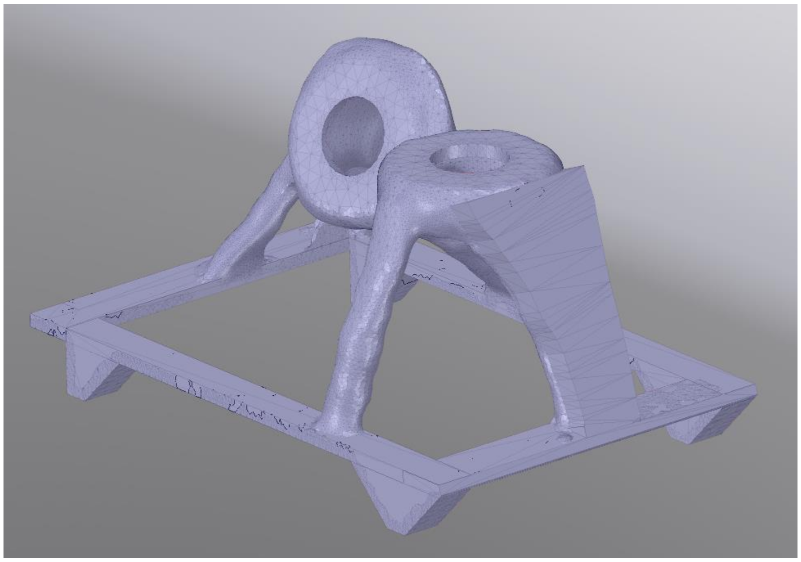


Figure 30: *The completed base, or foundation. Note that the previous changes have been combined into one solid part.*

Rev3e is corrupted and lost due to program crash. Rev3f continues working with the base. We are trying to mold the shape of the foundational platform into something that matches the geometry of the organic part. This can be seen in Fig. 31.

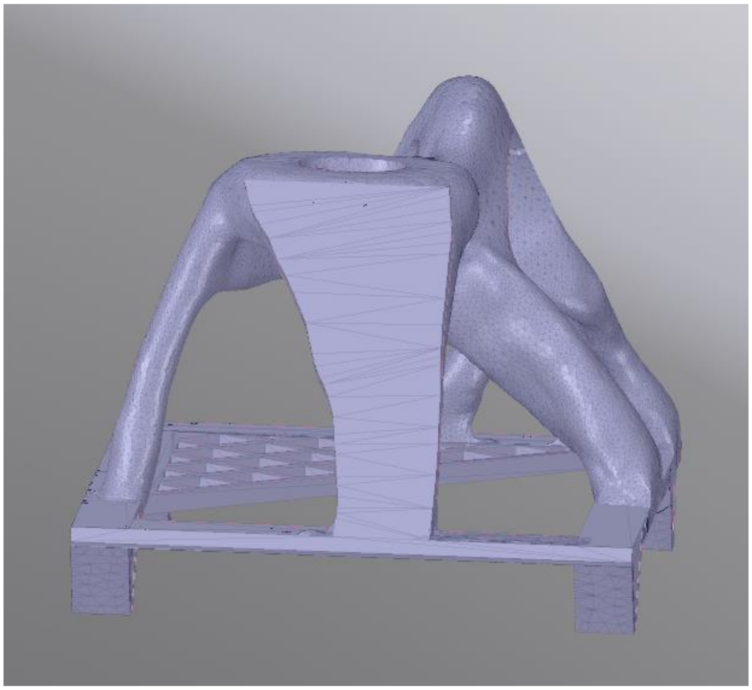


Figure 31: *Rev3f changes the foundational platform to more closely resemble the organic geometry of the part.*

Rev3g starts to address the mounting locations for the rocket housings. We can see in Fig. 32 the green circle that is giving these locations nice smooth locations for mounting.

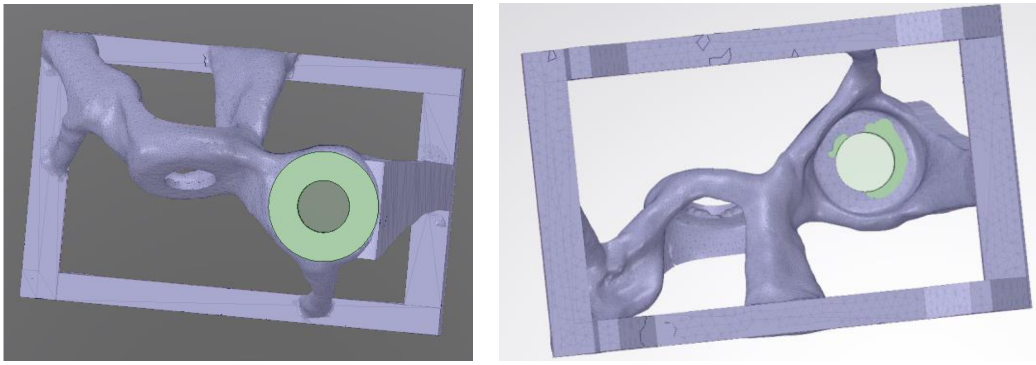


Figure 32: *Rev3g starts to correct the mounting surfaces for one rocket mount location. The left shows a picture looking down, the right shows a picture looking up.*

Revisions Rev3h and Rev3i correct the transition from the foundational platform to the rocket mounting location. We can see this change in Fig. 33.

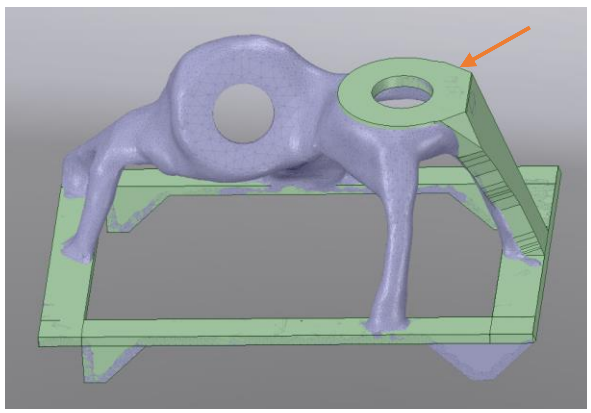


Figure 33: *Rev3h, and Rev3i combine the foundational platform with the smooth mounting surface. This location can be seen at the location of the arrow.*

Rev3j corrects the mounting location for the second rocket. These changes can be seen in Fig. 34.

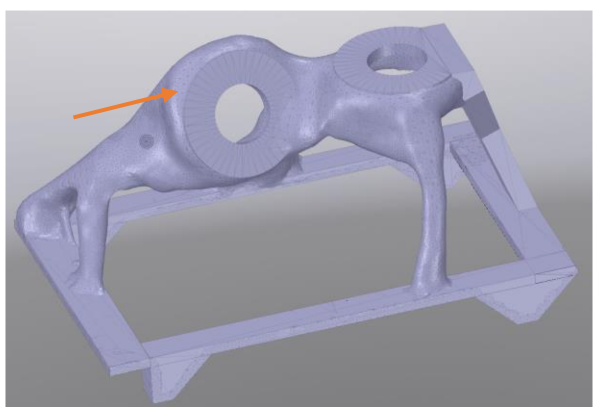


Figure 34: *Rev3j corrects the mounting location for the second rocket. Notice the extremely well-defined circular top where rocket two would mount. This can be seen at the location of the arrow.*

Rev3j also changes the foundational base. The organic structure that existed prior to the base has been removed, and only the added geometry remains. The shape of the foundational platform has been modified as well for printable angles. This can be seen in Fig. 35 and 36.

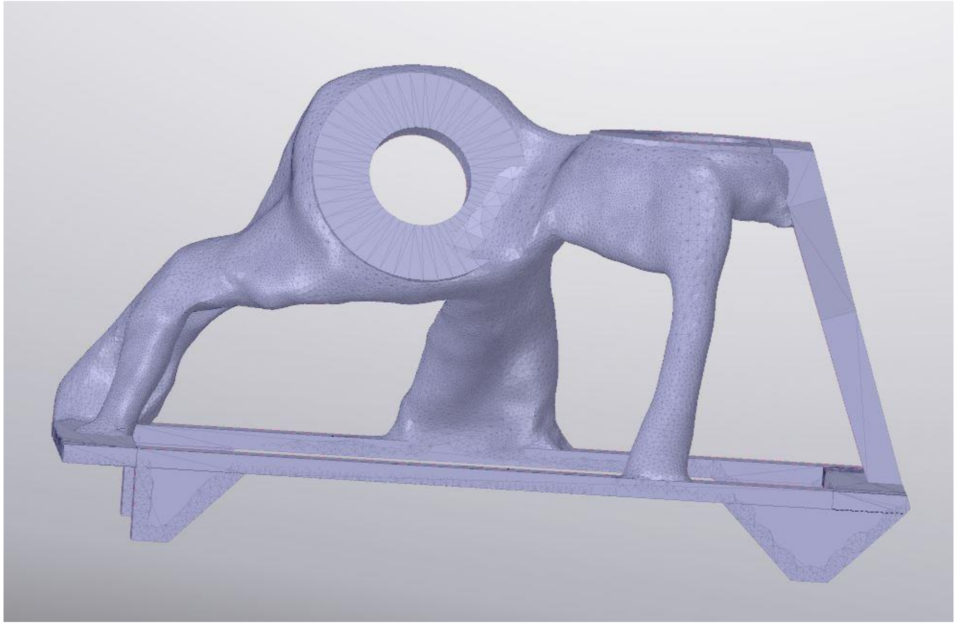


Figure 35: *Rev3j removes the organic shape that existed where the foundational platform was added*.

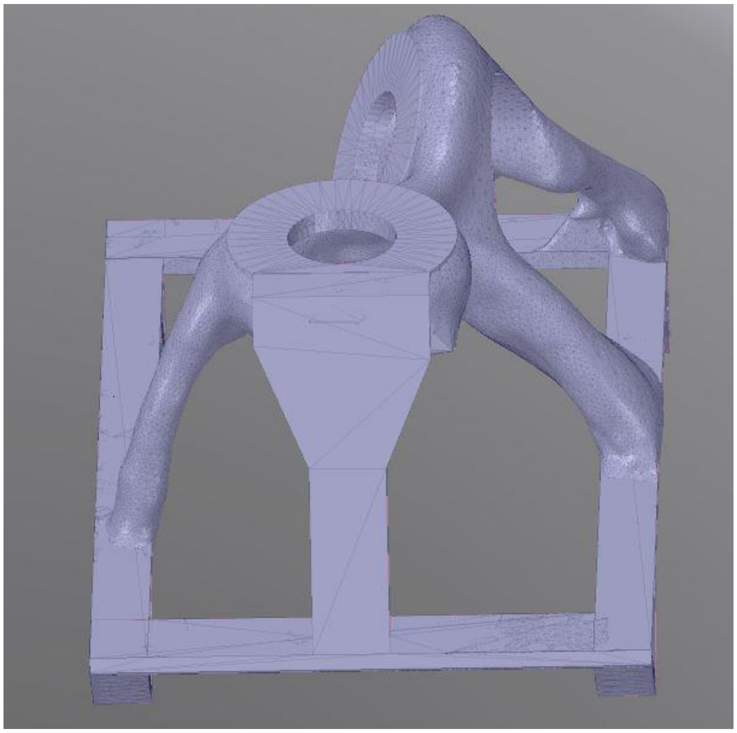


Figure 36: *Rev3j changes the shape of the foundational base.*

Rev3k, Rev3l, and Rev3m use mesh mixer to smooth out the rough organic features of the model. These can be seen in subsequent revision pictures.

Rev3n introduces a large triangular support. This triangle is used to support the entire upper portion of the bracket to eliminate support material generation from the base to the top. Weight relief diamonds are cut into the triangle to make this addition as light as possible. This change can be seen in Fig. 37.

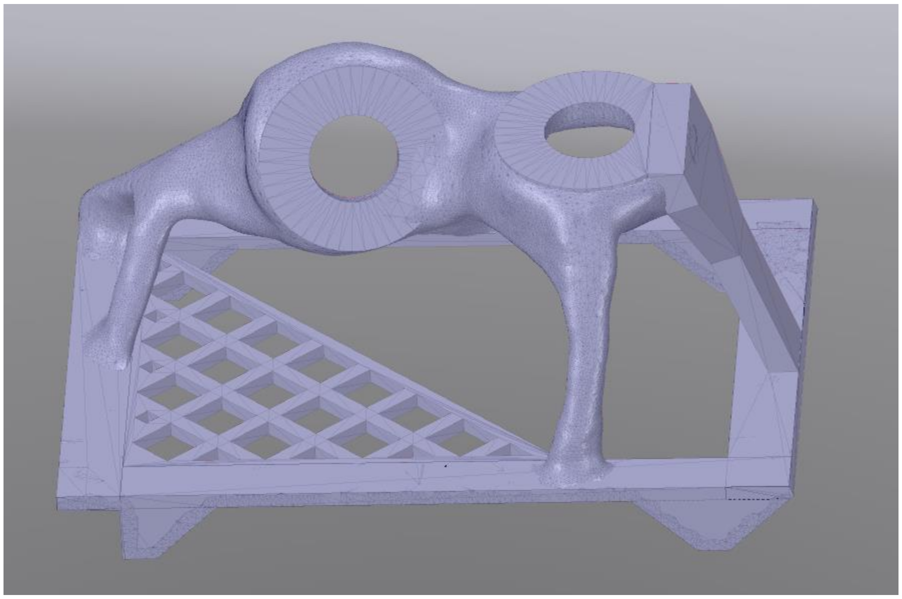


Figure 37: *Rev3n adds the large triangle to the structure. This will support the top of the bracket as it is printed to eliminate support material. Diamonds are cut into the triangle to reduce weight.*

Rev3o is a backup save.

Rev3p adds a thin layer of material to the foundational plate. This extra material allows for the part to be cut from the build plate with the band saw. This can be seen in Fig. 38.

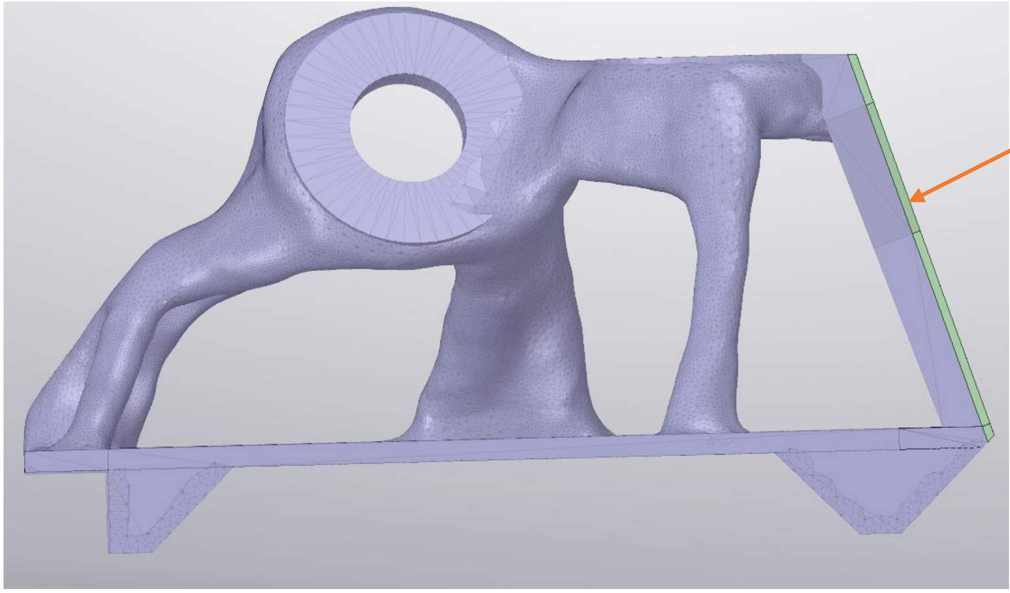


Figure 38: *Rev3p adds sacrificial material to allow the part to be cut off from the build plate. This can be seen at the location of the red arrow. The added material is green.*

Rev3r, Rev3s, Rev3t, Rev3u and Rev3v add gussets to the base of the organic arms. This will ease the transition of the support material onto the organic arms. Each revision works on one individual gusset. Rev3v has all gussets added and can be seen in Fig. 39.

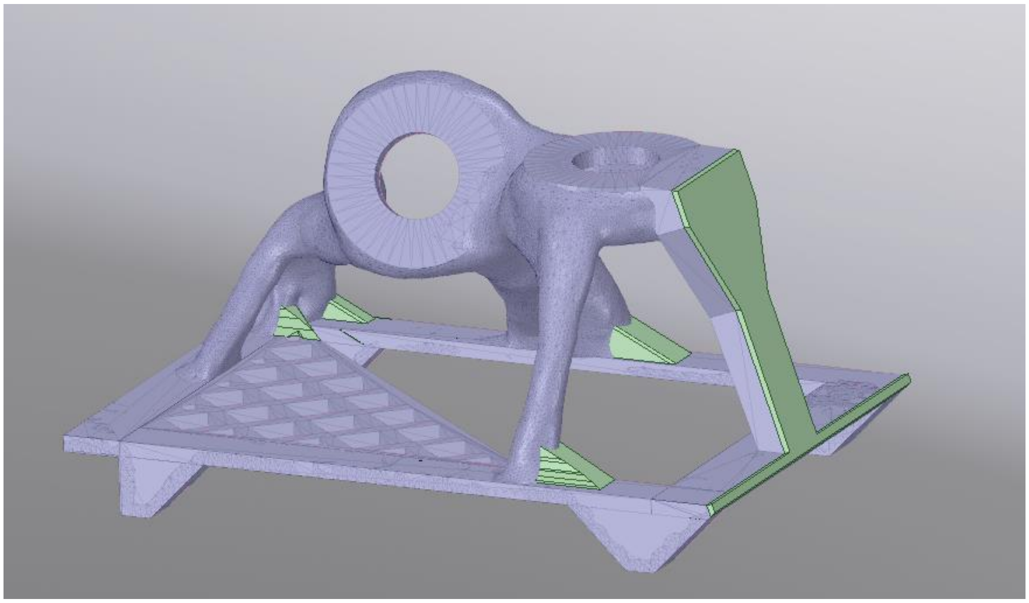


Figure 39: *Revisions 3r, 3s, 3t, 3u and 3v all incorporate gussets onto the part. These will ease the transition of the support material onto the organic arms. They can be seen in green.*

Rev3w turns the previous revisions into one solid part.

Rev3x is the final part, with the last smoothing done in mesh mixer. This can be seen in Fig. 40.

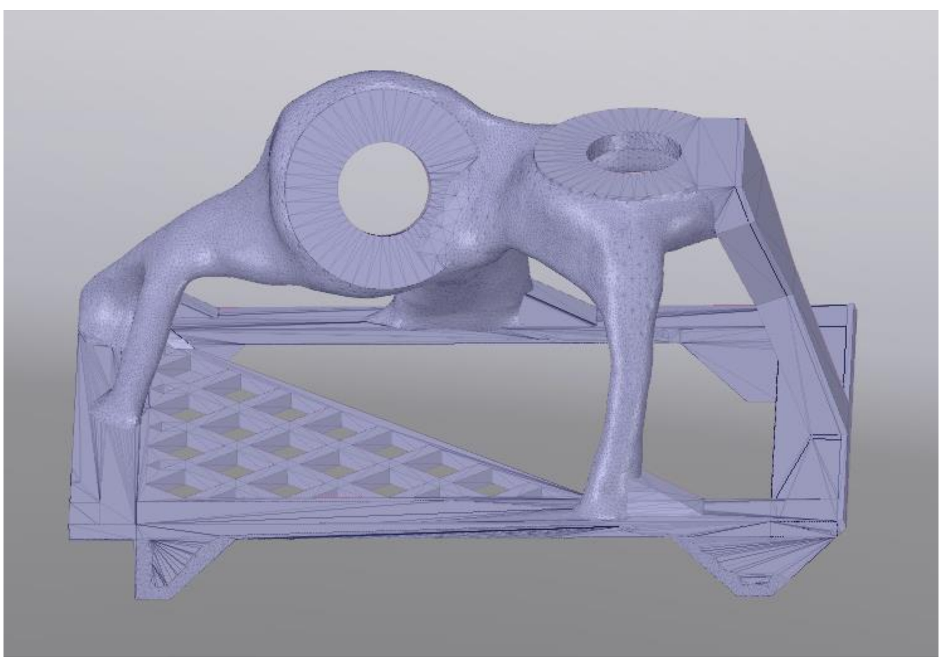


Figure 40: *Rev3x. The completed rocket housing.*

The side pieces underwent a tremendous amount of modification and are almost unrecognizable from the previously shown side pieces. The reason this happened is that they were further and further refined by performing topology optimizations followed by a modal analysis to determine the natural frequencies, which needs to be greater than 100 [Hz] due to launch loading vibrations. Once the natural frequency converged onto the limit, the part was no longer modified with topology optimization. This allowed for a tremendous amount of weight to be removed. Figure 41 shows the side piece before further modifications.

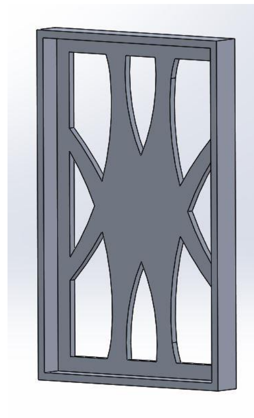


Figure 41: *Unmodified side piece.*

Unfortunately, we did not save each change as a revision to the side piece. We simply continued to remove material based on the results of the finite element analysis and modal analysis. Once we had reached a shape where no more material could be removed, we cleaned the part up manually, giving it recognizable geometries. This was done so that the part could be traditionally machined with minor changes, such as removing sharp corners and replacing with a reasonable radius. Finally, weight reduction diamonds were added, and an angle was introduced where the part touched the print bed of the printer. This was done so the part would be below the printer's maximum height restriction. The side pieces are within design specifications for stress and frequency and can be seen in Fig. 42.

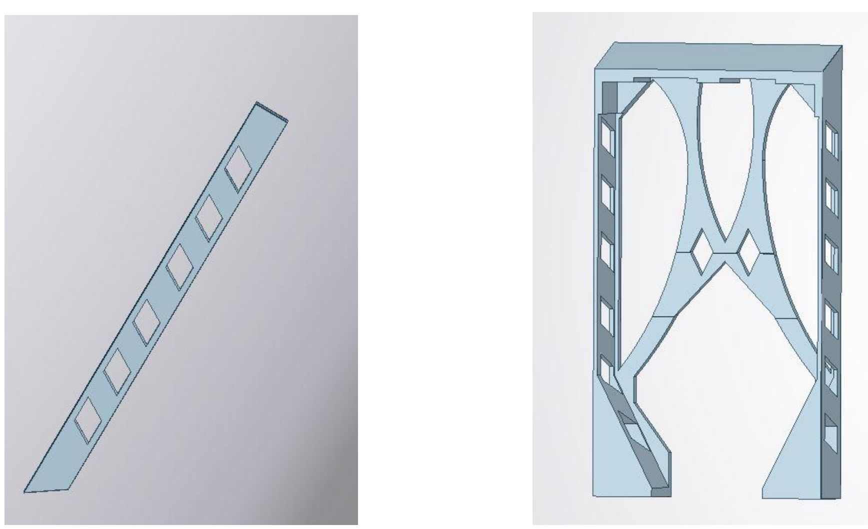


Figure 42: *(Left) the side piece in its print orientation. Note how the angle at the bottom allows the part to be printed at a printable angle while staying under the maximum build height of the printer. The weight relief diamonds are also adjusted to be at printable angles. (Right) The side piece as it would stand supporting the rocket housing. Note the angled surface at the top that would need to be machined off for a mating surface. The areas of greater material thickness allow for bolting locations. The side pieces are within the design limits of stress and frequency.*

The stress, strain and modal analysis can be seen in Fig. 43, 44, and 45.

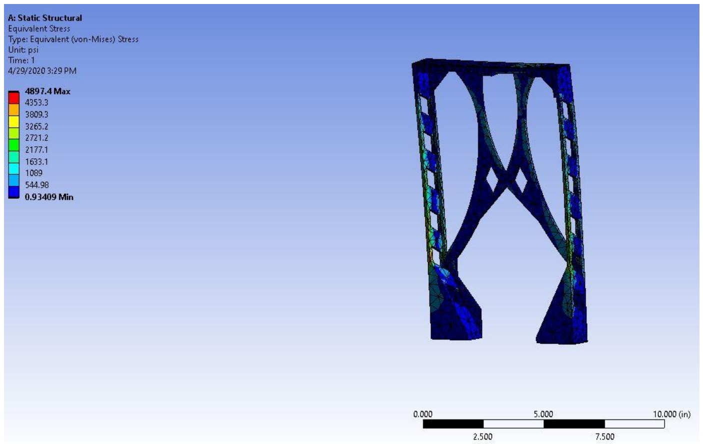


Figure 43: *Stress analysis on the side piece.*

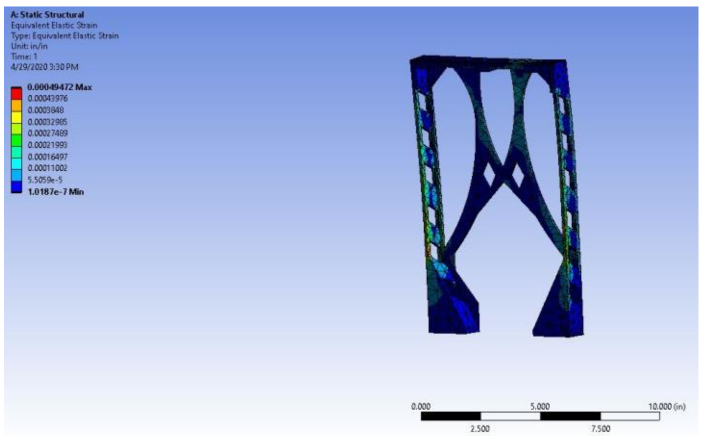


Figure 44: *Strain analysis on the side piece.*

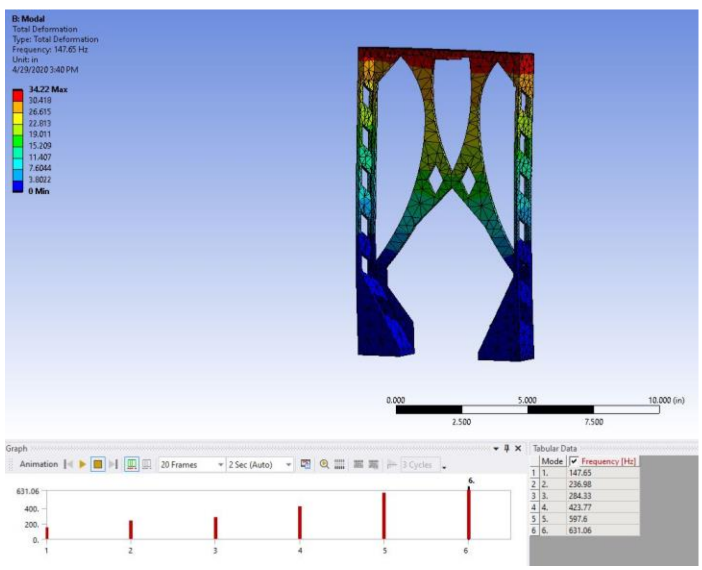


Figure 45: *Modal analysis of the side piece to determine natural frequency*.

The valve plate that was designed to accompany the generatively designed bracket had three main considerations to be taken into account. The first was the ability to withstand the stress of launching into a high Earth orbit, the second was to have a natural frequency greater than 100 [Hz] in any of the ways that a vibration could occur, and the last design consideration that was taken into account was channeling heat to the service valves. The elements that are carried to each iteration of the design are the center “X” shape and the squares where the valves mount.

We began the design with the locations of the four valves because their placement could not be moved. The original shape produced can be seen in Fig. 46. The way that these four areas where connected was with an “X” shape. This has two advantages: it saves weight and means that the heater does not have to force heat through as much material. The reason to have all four areas in a square shape is that it does not have a front or a back and thus simplifies the machining so that the holes are the correct size. For this iteration of the plate, it was not known how it would mount to the overall assembly.

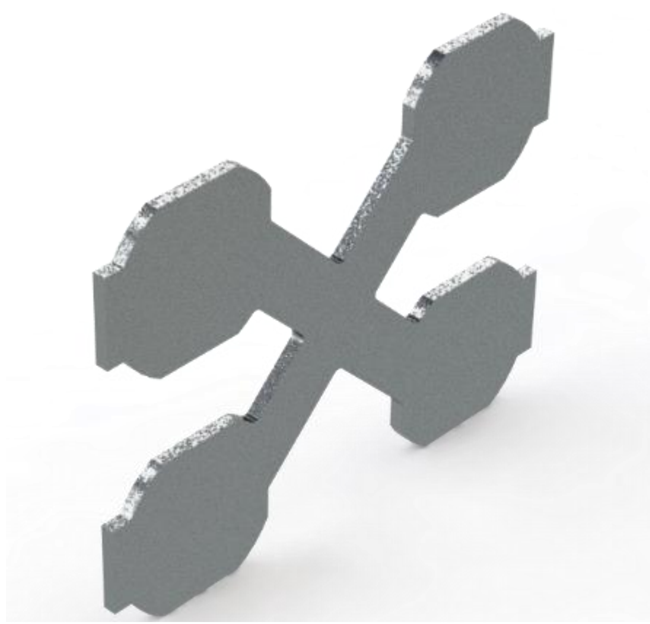


Figure 46: *Iteration #1 Valve Plate.*

The next step in the design process was to figure out how the plate would mount to the greater assembly of the bracket itself and to the satellite. The first idea that came to mind was the extension of the “X” shape so that it could reach the side pieces and the overhang of the edge of the satellite. At the end of each of the tips is a small flange for holes to be drilled into it so that fasteners can be used. This iteration of the plate is displayed in Fig. 47. The one worry for this design was the loading during the launch of the vehicle. The natural frequency was also a concern because the ends are quite thin. Upon analysis of the part it was found that the ends did not meet the natural frequency, with the lowest frequency being 80 [Hz]. This did meet the loading requirements.

Figure 47: *Iteration #2 Valve Plate.*

The next iteration aimed to stiffen the ends of the "X". This was done by adding struts to all four of the sides, which solved the problem of the natural frequency that the previous iteration had. The lowest frequency for this iteration occurs when the center section is in vibration, which is often called the drum skin loading. This frequency is approximately 250 [Hz], which is an effective factor of safety of 2.5. All other frequencies that were computed where greater than the drum sink by a factor of 2. This met all other design criteria as well. With the loading and natural frequencies taken into consideration, the number of fasteners was starting to be examined. The first form of this iteration had 12 #8 bolts to mount the plate to the bracket assembly and 4 #8 bolts to mount the satellite vehicle for a total of 16 fasteners used. This amount was deemed excessive, especially for mounting the valve plate the rest of the bracket. It was then studied in order to figure out the minimum hardware needed for both the bracket and vehicle mounting. The number decreased from 12 to 4 for mounting the valve plate to the bracket, and from 4 to 2 for the bracket to satellite connection. A comparison of the final designs is shown in Fig. 48.

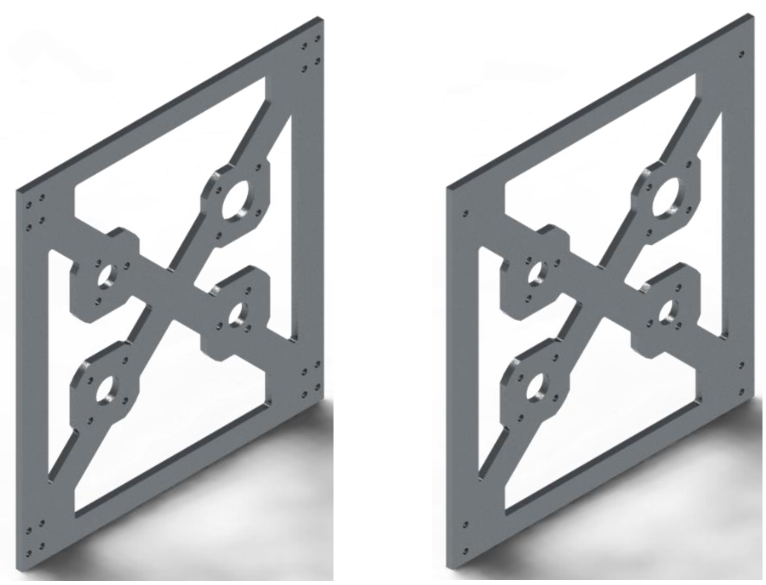


Figure 48: *Final Valve Plate with Original Mounting (left) and optimized (right).*

The complete Ansys bracket uses 3 unique parts: 1 rocket housing, 2 side pieces, and 1 valve plate. Figure 49 shows all parts in an assembly.

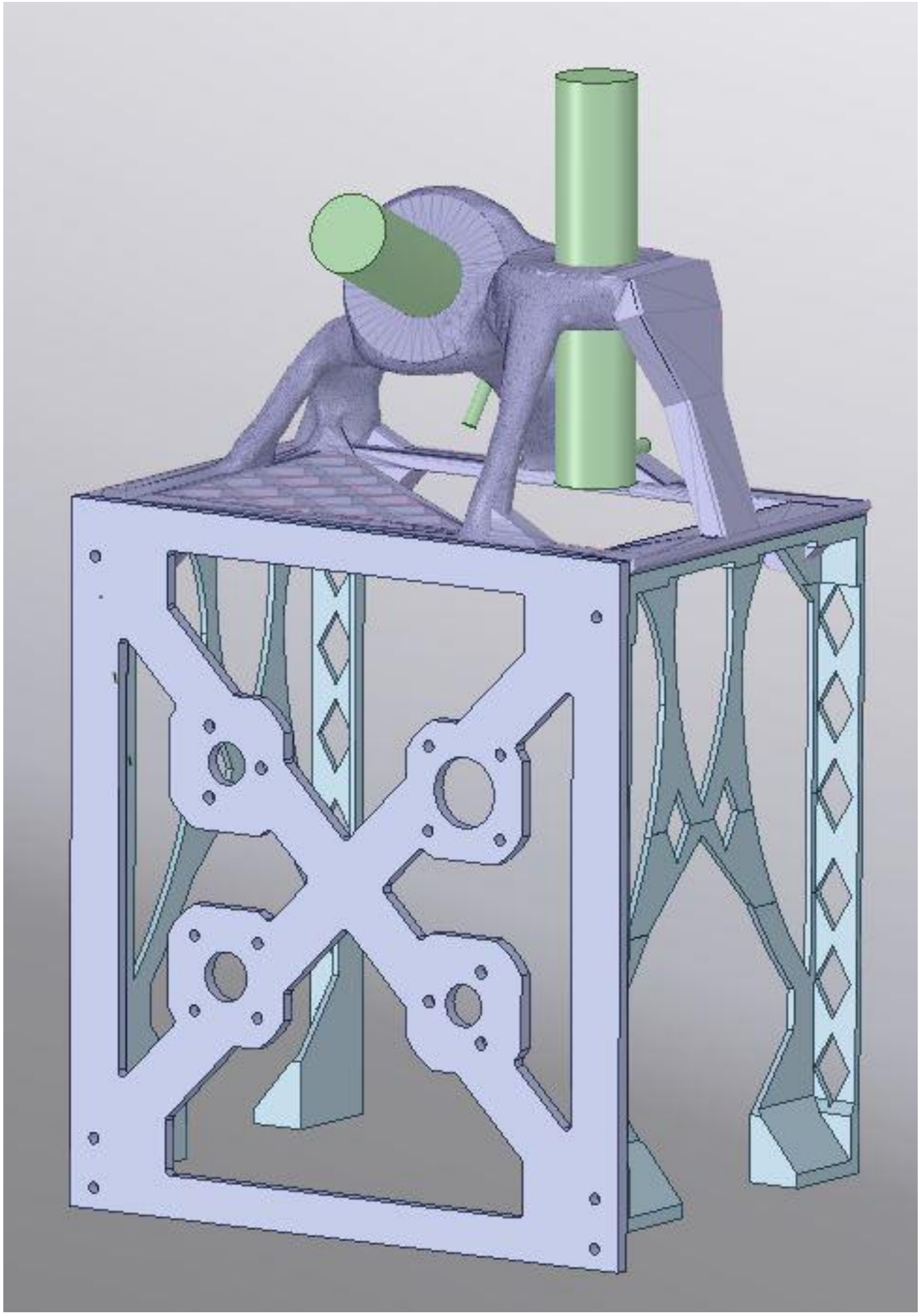


Figure 49: *The 4-piece rocket bracket assembly.*

The locations to secure the components together were built into the parts. Selecting the hardware had yet to be determined when world events pushed the project off course. The hardware locations can be seen in Fig. 50.

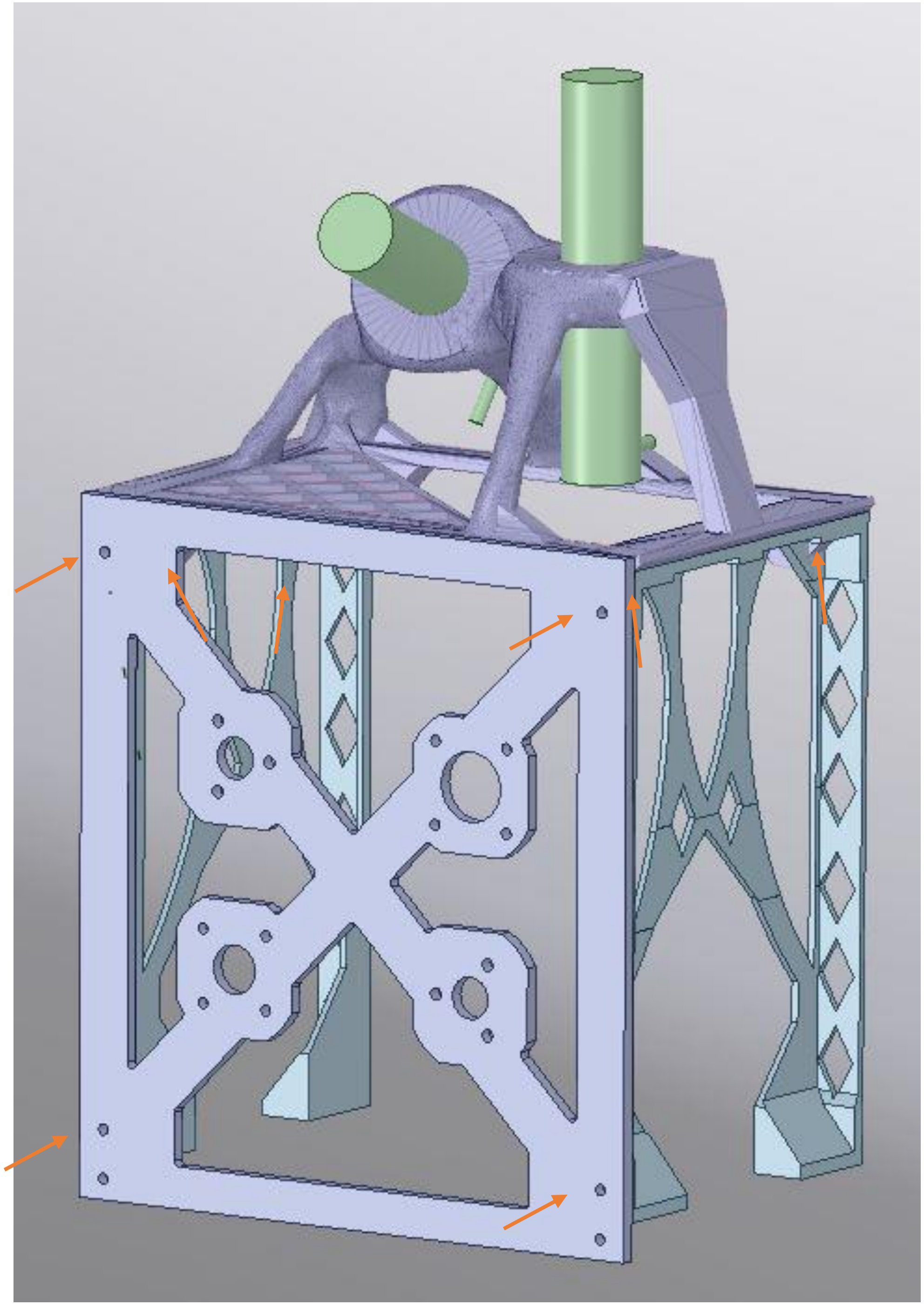


Figure 50: *Bolting locations shown with arrows. Note that two bolt locations are not visible in this assembly orientation but are identical to the side piece bolting locations seen on right side of picture with two upwards arrows.*

The assembly has two extra holes on the valve plate that are below the bottom edges of the side pieces. These are there as a representation of bolting the valve plate to the GPS III satellite.

Further work on this system could include modifying the rocket housing to incorporate an anti-capture design. This would allow the rocket housing to be removed if the rocket thrusters were already installed. This addition would protect completed work if parts were damaged. The rocket housing would undergo weight reduction as well, and bolt sizes would be selected. Finally, we would print the prototype parts to conduct experimental modal analysis.

The satellite support system bracket assembly was designed with a combination of a generative design approach, and topology optimization. These design approaches allowed us to produce parts that met stress, part reduction, fastener reduction, and frequency constraints. With more time, we would also be able to reduce mass even more. This is a successful bracket not only because it meets the constraints, but because it shows what is possible when considering additive manufacturing as the manufacturing method.

Software Evaluation

While we were stopped from actually testing some of the parts to confirm their validity (after all, a part could pass simulations and fail in actual testing), we still had a way to evaluate each software. Throughout the semester, our team had been in contact with Bob Rauscher, who we worked with to define a set of criteria we could use to compare the software packages to each other. We then ranked the individual packages, on a scale of 1-5 (with 1 being best possible, 5 being terrible/nonexistent), to determine which one was superior. Table 3 demonstrates which software was better in which regards.

Table 3: *Ranking of Ansys and nTopology Platform in terms of various criteria*

Golf Rules For Scoring, lower is better	Ansys	nTopology Platform	Best in Each
User Interface(Beginner user)	3	4	Ansys
User Interface(Advanced user)	2	3	Ansys
Graphical representation of parts	2.75	2.75	Ansys and Platform
CAD File Import and Export Ability	1.5	2.5	Ansys
Ease of Importability for CAD files	1	2	Ansys
PTC Creo Plugin	1	5	Ansys
Workflow	2	1.5	Platform
Finite Element Analysis	3	3.25	Ansys
Ease of using advance features (i.e. FEA Topology/ Generative features)	2	2.75	Ansys
Generative Design/ Topology Optimization	2.5	2.5	Ansys and Platform
Lattice Formation	4.5	2	Platform
Post optimization design time to print part	5	5	Ansys and Platform
Printability of Volumetric Lattice Formation Parts	5	2	Platform
Numerical representation of results	2	3	Ansys
Graphical representation of results	2	3	Ansys
Additive Manufacturing Integration	5	2	Platform
Software Uptime first time input (User Input Time)	2	3	Ansys
Post Processing Input (User Input Time)	4	4	Ansys and Platform
Software Downtime (Simulation, Updates, and	3	4	Ansys
Average	2.80	3.01	

The details of each criteria were specified as follows:

- User interface (Beginner user): This was a way to rank how approachable the software was at the beginning, and how comfortable we felt doing things in an effort to self-teach. We found Ansys here to be slightly more beginner user friendly, given its clean, familiar menu design.

- User Interface (Advanced user): This was a metric set to how friendly the software was when trying to do advanced techniques, now that the basics had been worked out. Again, Ansys edged out nTopology, in large part thanks to a user interface that retains a consistent look across the project, and a plethora of tutorials and online videos that are easy to follow and repeat.

- Graphical Representation of parts: How well does the software portray the part you are making. In both cases, the software packages lacked any kind of real view or rendering capability. In addition, the sheer number of faces makes rendering the parts on 3^{rd} party software very difficult, essentially pigeonholing you into accepting a slightly cartoonish model.

- CAD File Import and Export Ability: Put simply, how well can the program read other cad files, and how many files can it save in. Again, Ansys took the edge here, with it being able to work and save in more formats than nTopology.

- Ease of Importability for CAD files: Essentially, is it easy to import and export, or not? Neither software had issues working in different file types, but seeing as Ansys had more file types, the win goes to Ansys.

- PTC CREO plugin: Lockheed Martin uses PTC Creo for its CAD needs, and having a way to keep native formats between the two software's would save a lot of time. Ansys has a plug in that allows this, and nTopology does not. A note: For our designs we utilized SolidWorks, as it is a program with which we are all familiar and have full licenses for. However, reviews of the plug in are positive, and we would anticipate no problems for Lockheed Martin should they use Ansys.

- Work-flow: This one is kind of vague and generalized, but the idea was, once you feel comfortable in the software, do you feel comfortable following an established workflow, or are you always trying to figure out how to get it to work for you. Luckily both packages had good workflows, and nTopology, with its easier to follow interface, took the win here.

- Finite Element Analysis: This was checking both how easy it was to do an FEA on the part, but also how well we trusted the results. Ansys won here by a hair, as it is a slightly smoother process, with more options to verify your simulations.

- Ease of using advance features (i.e. FEA Topology/generative features): These features are difficult to execute and get good results from. In this case, Ansys won, as there are a lot more resources online to walk you through how to implement these features, and you are not stuck just trying out new things and hoping they work.

- Generative Design/Topology Optimization: Although different processes, we wanted to see how we felt about their performance as a whole. Essentially, did we feel like they were able to execute their purpose, or did they feel like they were not quite getting to pure optimization/pure generative design. While both are difficult to use, have flaws, and most importantly were not able to be verified with physical parts, we felt both of them performed about the same. Room for improvement, but we felt good about what we had after learning how to use the software and refine the designs.

- Lattice Formation: Ansys can create lattices on parts, but it is nowhere near as convenient and approachable as nTopology's method. For creating lattices to reinforce parts while adding minimal weight, nTopology wins hands down. Given lattices are a great easy way to update and improve existing designs, we see this is a big win for nTopology.

- Post optimization design time to print parts: After the part was created in each software, how reasonable was the amount of time it took to refine the part to be printed. Here, both software's earned 5's- neither of them gives you a part you can instantly print out of metal. The feature simply does not exist, so there is a lot of post-design time that goes into taking these designs from the computer screen to real life. This is naturally a problem, but one we anticipate both companies are working to solve.

- Printability of Volumetric Lattice Formation Parts: nTopology's lattices printed with minimal effort, which is another big win for nTopology. With Ansys, we were never even able to attempt a test, but the lattice formation was already much more difficult.

- Numerical Representation of Results: How well can you read the data for simulations in the software's. This is a critical component of software evaluation, as being able to clearly not only see pictures of the simulations, but to quickly get the results tabulated saves invaluable time for engineers. Ansys not only provided clear pictures, but it also gave us clearly defined numerical representations, which made it much easier to compare between designs.

- Graphical Representation of Results: How well does the data show up graphically? Again, Ansys was slightly more polished, and therefore you could interpret the results more quickly and clearly.

- Additive Manufacturing Integration: While neither software gives you a printable design right out of the gate, nTopology has done more to integrate information from readily available printers than Ansys. nTopology can then see this data to inform the design. Although there is still work to be done once the model is made, nTopology is able to take printer considerations into account earlier than Ansys, resulting in less reworks.

- Software Uptime (first time user): On a scale of 1-5, how much time did it take the user to create a part when they were still novices. Both software's were fairly approachable, but with so many more online tutorials and resources, Ansys won out. By the end, when all users could be considered advanced, the time spent from start to end of design was fairly comparable.

- Post Processing Input (User Input Time): How much time did it take for a user, after the design was ready, to get the part ready to begin post processing. Again, both software's took a lot of time to move onto the next steps, thanks to the sheer size of the models.

- Software Downtime: When you were letting the software execute a task (updating, running simulations, etc) where the user did not have to be present, how quickly could the software produce results. The more refined Ansys, once again, won here as it requires less input and less time to get useable results from simulations, and its updates are all in the background and can be set automatically.

By examining the table, the software preference is clear. We have high hopes for nTopology - they are constantly improving, and just in the eight months that we have been working with the suite, we've seen massive improvement. However, given Ansys' higher levels of refinement and larger berth of available capabilities, Ansys was shown to be the clearly better solution of the two design software packages.

Ansys is easier to learn, and with a static UI, tutorials made from previous versions are still relevant and easy to follow. What is more, with all of its evaluation capabilities, Ansys is capable of handling more strenuous designs, and the parts made in Ansys can be evaluated in a multitude of ways to validate any design you make.

nTopology still has its benefits, of course. The lattice formation is a quick and easy way to start updating old designs. In addition, the ability to use topology optimization means that for Lockheed Martin, nTopology could begin making improvements relatively soon, as it could begin optimizing existing parts with little to no effort.

Both design software packages offer their benefits, and we can see a place for both at Lockheed Martin. The reality is that the AM landscape is changing very quickly, and it is hard to anticipate exactly which method or technology will be the best going forward. Physical prototypes would go a long way in understanding the usefulness of each technology and package, but barring those, we believe that we can use metrics and our own experiences to at least make a recommendation on preferred packages.

If you were to only choose one package, it should be Ansys, as it scored consistently better on the Lockheed Martin criteria, is more refined and polished, and will enable Lockheed Martin to pursue generative design quicker rather than sooner. Also, Ansys is much more approachable to a new user, which will make it easier to integrate into a team's tool chest without costly and time consuming self-learning. The plethora of available learning materials online would allow for ambitious engineers to quickly advance in their mastery of the program, and we would anticipate Lockheed Martin being able to see benefits from the software relatively quickly. Being able to self-teach was obviously crucial for our team and would likely be crucial for an existing employee trying to learn the program, so we think the berth of online resources is very important. In addition, Ansys has the analysis features, and many other features which can help for designs that are not additive manufacturing driven. This means that not only would Ansys allow for new methods of design, but it would also be useful in evaluation and optimization of existing parts, making it very worthwhile.

We also felt it would be in Lockheed Martin's interest to examine nTopology. Where Ansys is the future, nTopology may serve as a bridge to that future, by allowing parts to be optimized and changed in a faster fashion, leading to more immediate results. Setting up a workflow to optimize parts to be printed at Lockheed would likely result in a multitude of improvements across projects and would open up the door for even more exotic designs in the future.

This is as clear of a conclusion as our team is comfortable making, given our current limitations. Truly, we hope that Lockheed would see fit to examine both software packages, as both bring new and exciting features which up to this point have not been seen. However, were a single software pilot program necessary to be run, we would recommend Ansys.